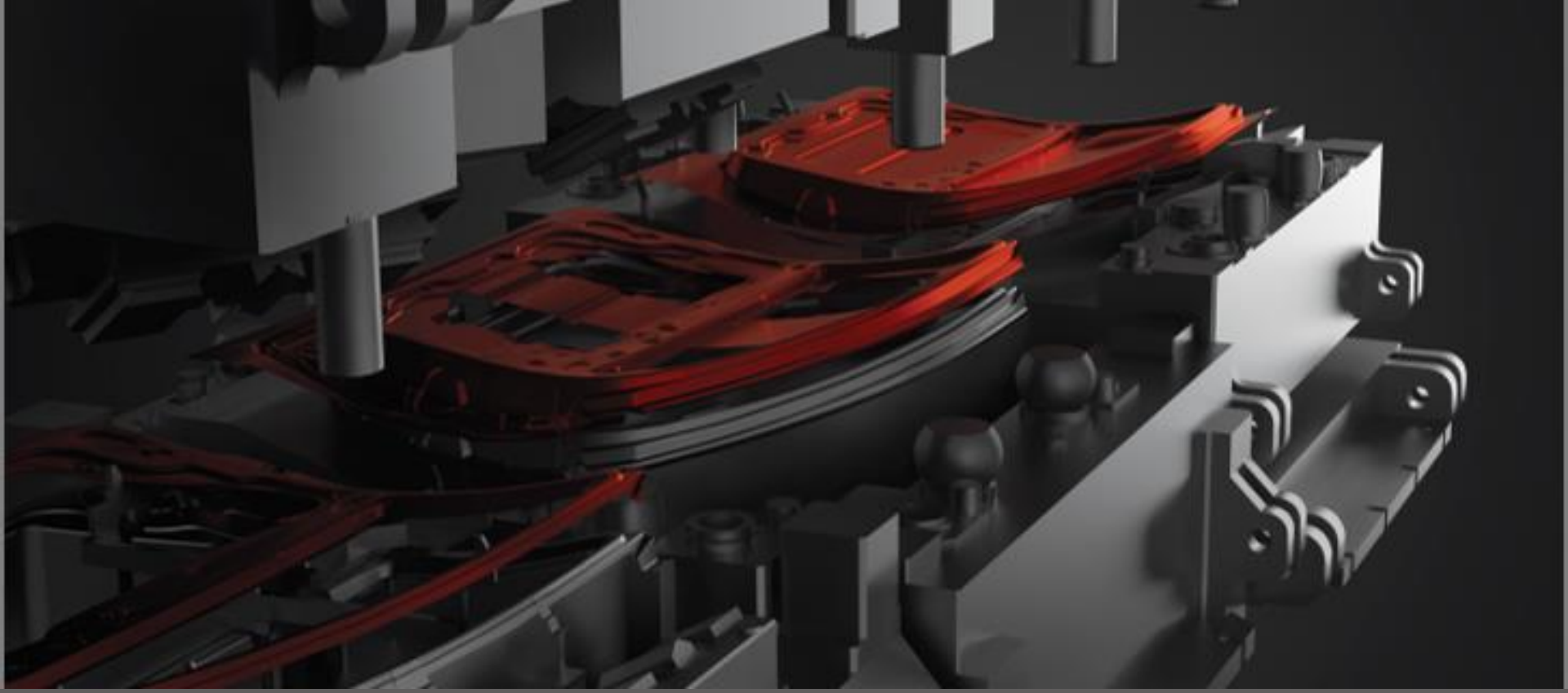


Inspire Form 실전 (3) - Stamping Material

한국알테어 • 김성문 기술사 • 2019-05-15

Altair

웨비나 순서

- Altair Inspire Platform
- Material Definition

Altair Inspire Platform

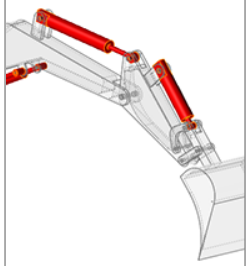
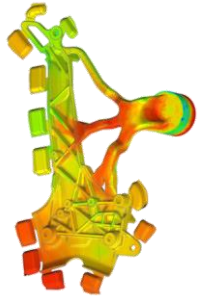
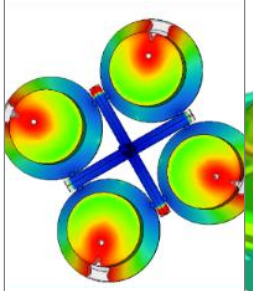
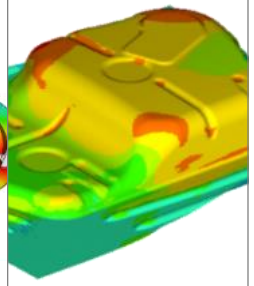
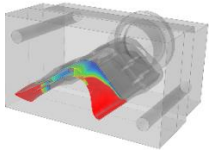
Altair Inspire - From Studio To Manufacturing

coming soon

coming soon

Studio | **Structures** | **Motion** | **Casting** | **Molding** | **Forming** | **Extrusion** | **Additive**



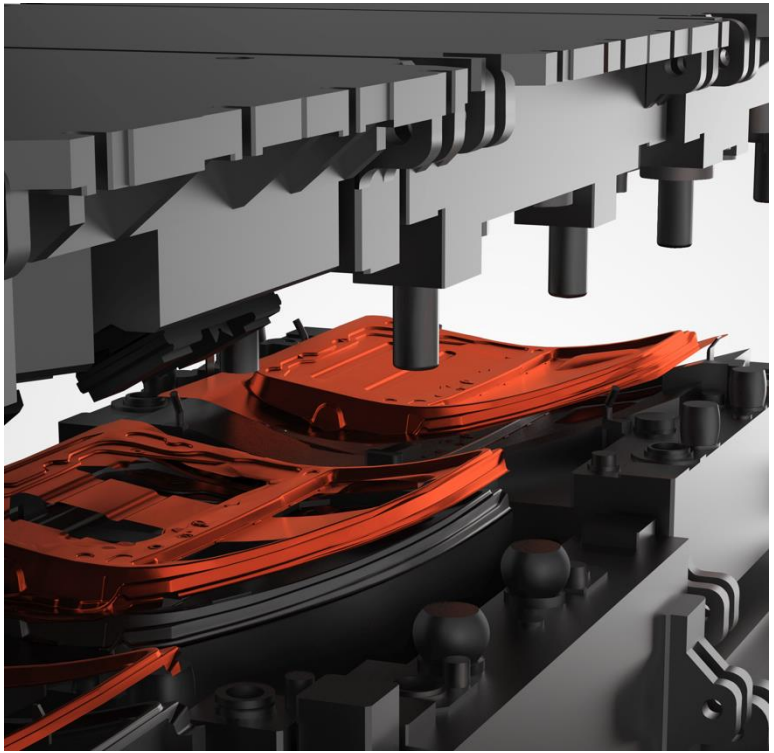







The industry’s most powerful and intuitive solution for design engineers to create high performing and manufacturable products.

What Is Altair Inspire Form?

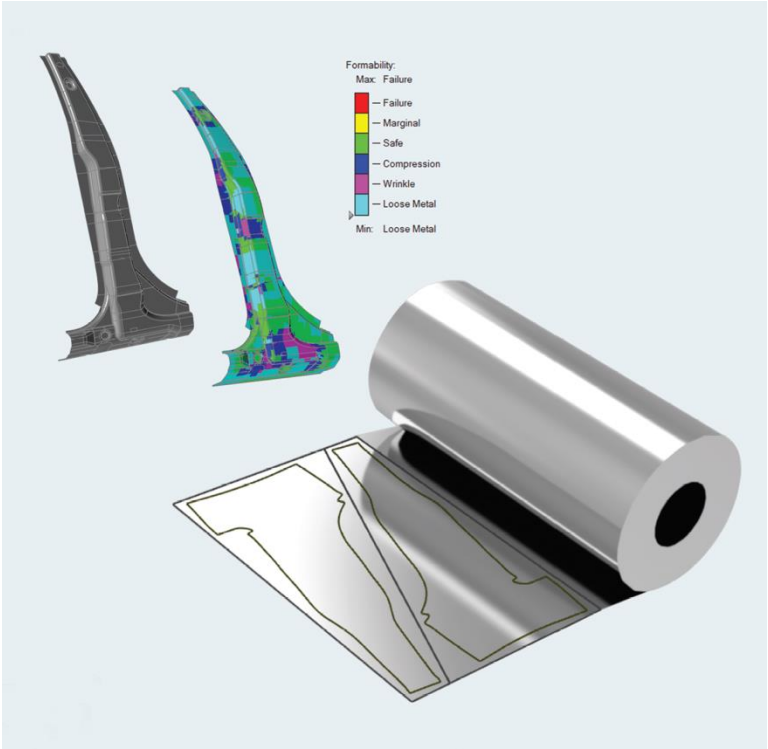
A streamlined stamping simulation software to

- Design better products with early check for forming feasibility
- Reduce scraps and improve material utilization
- Improve productivity with accurate formability analysis
- Optimize the process through single and multi-stage virtual tryout



Feasibility (1-step)

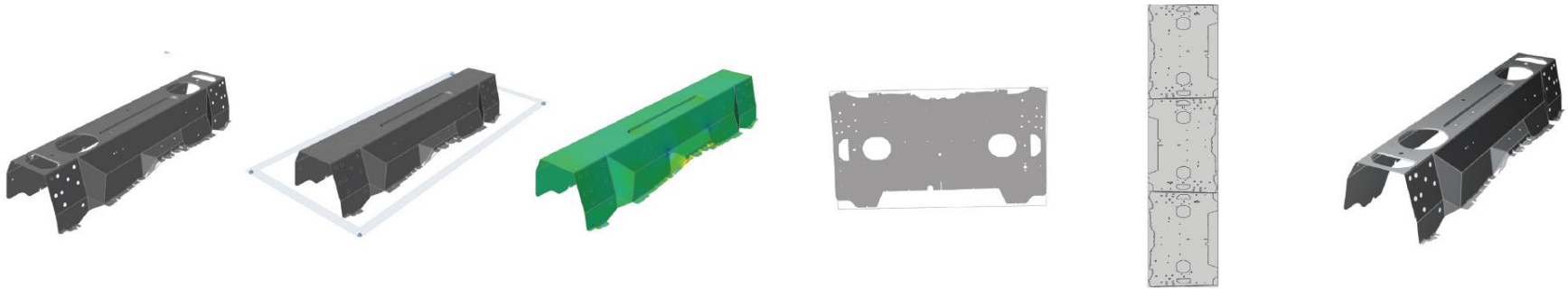
1. Use part CAD – no meshing required
2. Material database with US, Japanese and European standards
3. Automatic orientation to minimize undercuts
4. Apply blank holding force and drawbeads
5. Visualize thinning, formability, strains
6. Automatic blank fit and nesting to save material cost



Feasibility Workflow

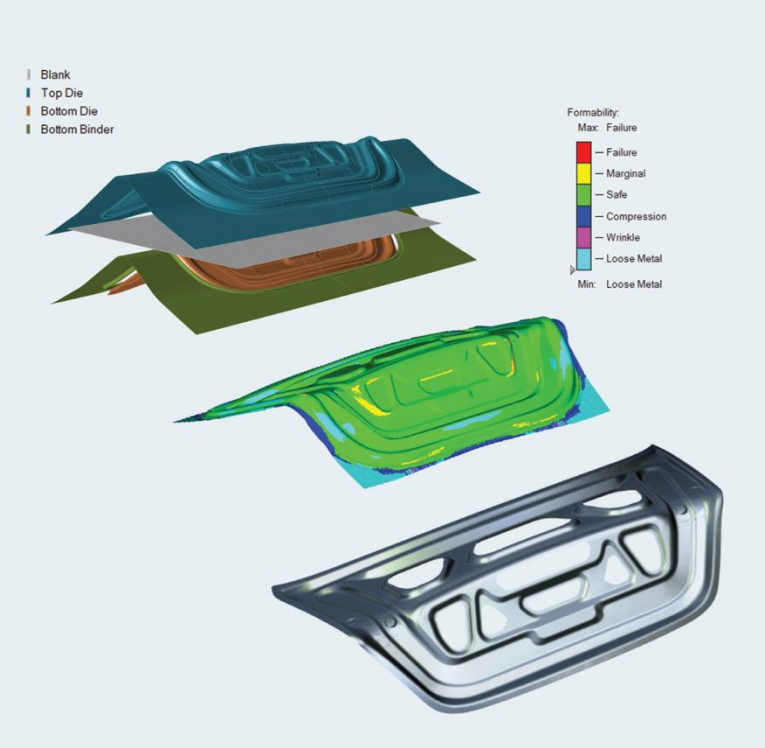
Quickly and reliably check the formability of the part early in the design phase by predicting potential for splits, wrinkles and loose metal.

The geometry of the flat blank can be automatically fitted inside regular shapes such as rectangle and trapezoid as well as nested on a sheet coil for maximum material utilization.



Virtual Tryout (Incremental)

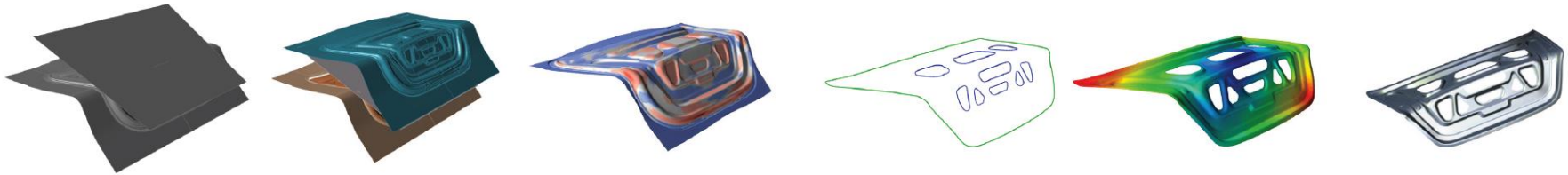
1. Intuitive and efficient environment for single and multi-stage stamping simulation.
2. Highly scalable solver for accurate forming and springback analyses
3. Thinning, strain, stress, failure contours and Forming Limit Diagram
4. Plot of tool force versus stroke
5. Callouts to view results history at any location



Tryout Workflow

Intuitively setup complex stamping processes including gravity, draw, redraw, flanging and springback with all tools and blanks.

Iterate on blank holder force and drawbead location and force to improve part quality.

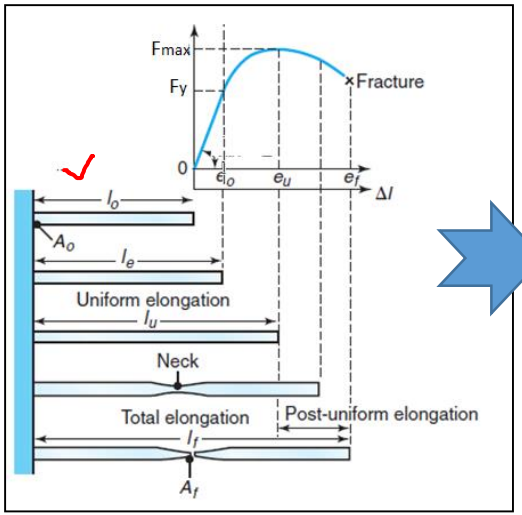


Import Model | Set Up Drawing | Analyze Drawing | Set Up Trimming & Springback | Analyze Springback | Form

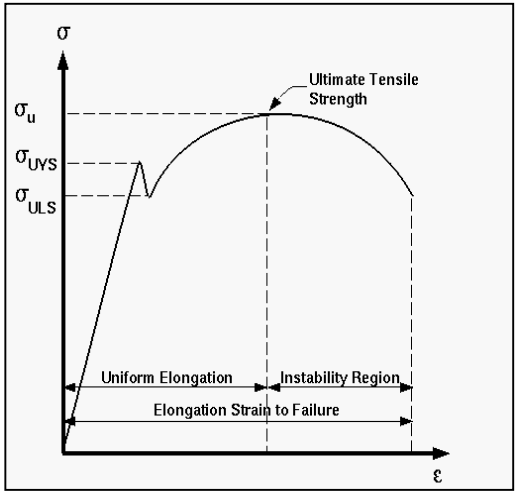
Material Test

Sheet Material Behavior

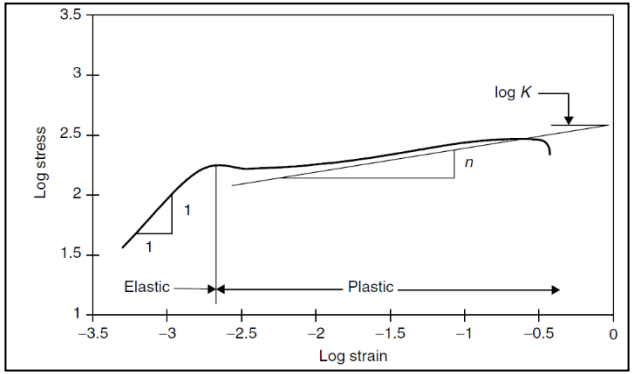
- **Sheet Material Behavior - Tensile test**



Force Vs Deflection



Stress Vs Strain curve



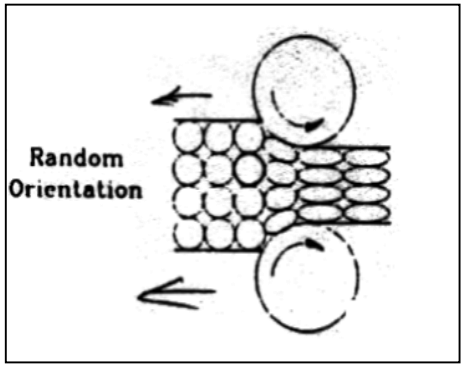
Fit to standard equations

$$\sigma_T = K\varepsilon_T^{\,n} \quad \text{or} \quad \sigma_T = \sigma_o + K\varepsilon_T^{\,n}$$

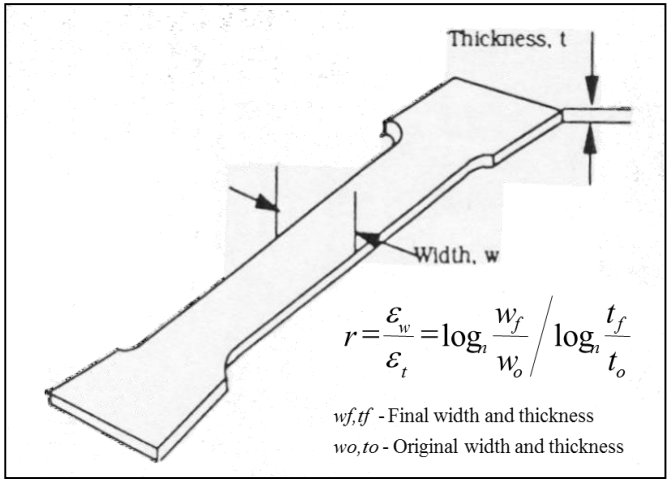
$$\sigma = K\left(\varepsilon_o + \bar{\varepsilon}\right)^n$$

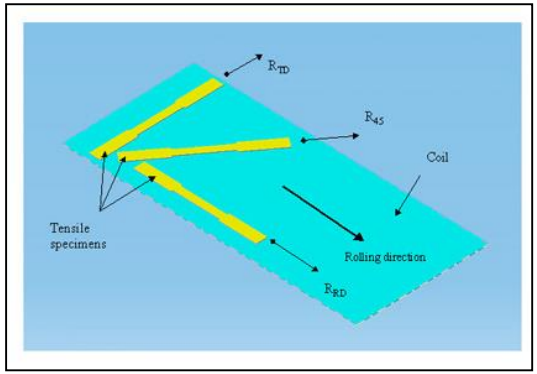
Sheet Material Behavior

- **Sheet Material Behavior - Anisotropy**



Grain oriented along rolling





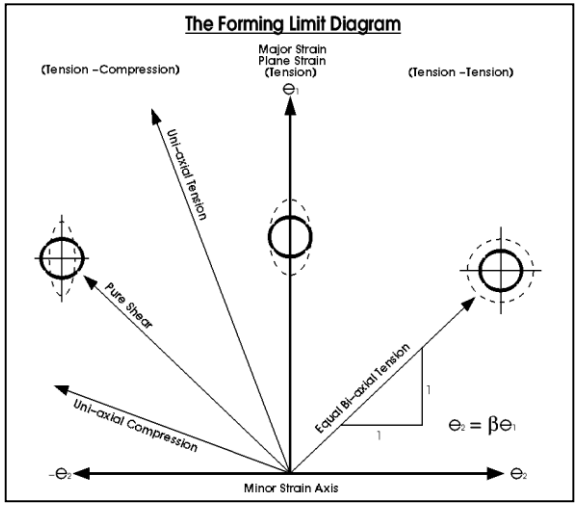
Rolling process causes orthotropy in sheet metal

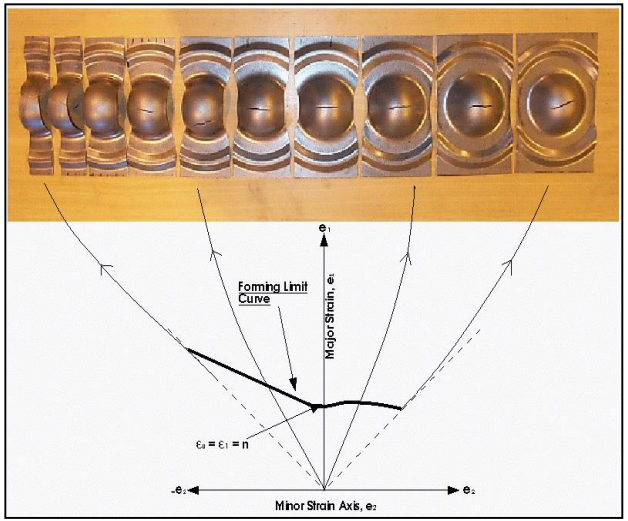
Plastic strain ratio R from Tensile test

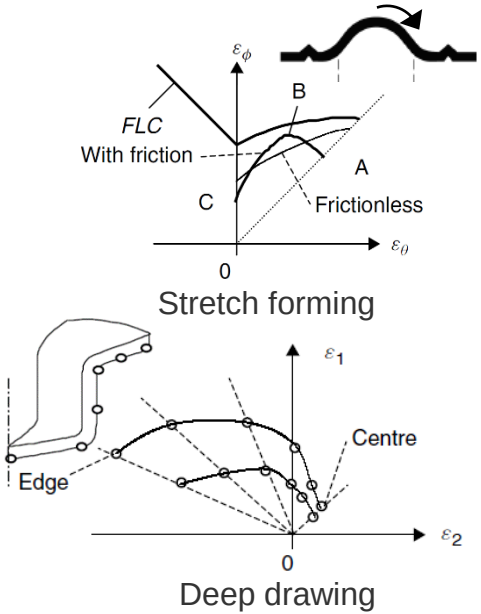
Quantify anisotropy in sheet metal

Sheet Material Behavior

- **Sheet Material Behavior - Forming Limit diagram**







- A plot of Major and Minor strain
- Location on the plot gives information of mode of deformation

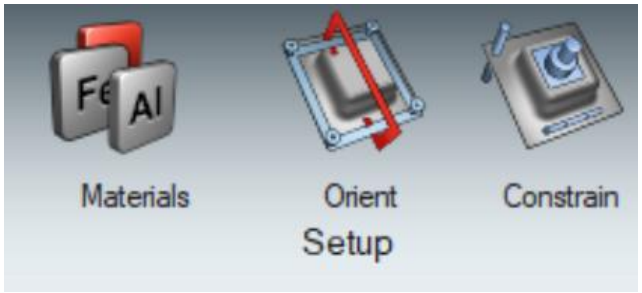
- Limit curve obtained from experiments gives the forming limit of material beyond which it fails.

- Captures all deformation modes

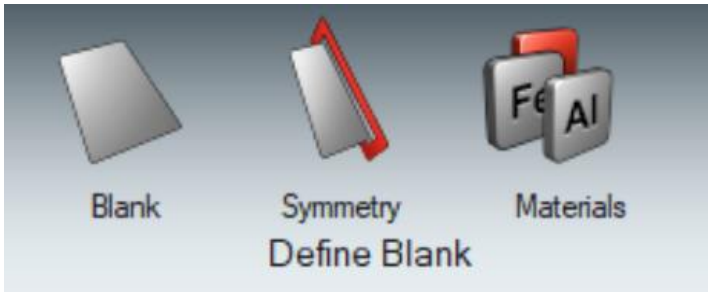
Inspire Form Material Definition

Feasibility / Tryout: Materials

- **Materials -** Create a material that is based on an existing predefined material in the System Library.
- **Orient -** Use the Orient tool to orient the stamping direction for a part.
- **Constrain -** Use the Constrain tool to create pin, drawbead, and blankholder constraints for analysis.



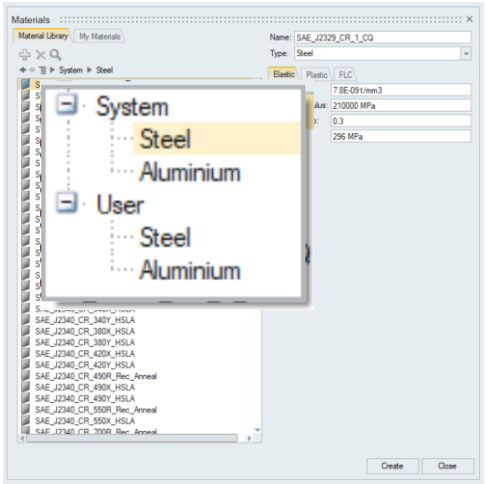
✓ **Feasibility**

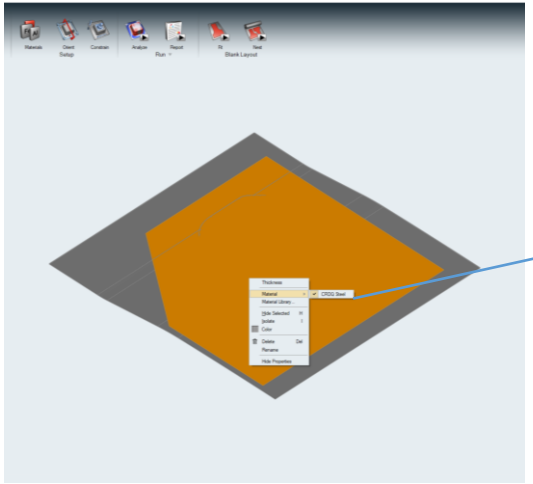


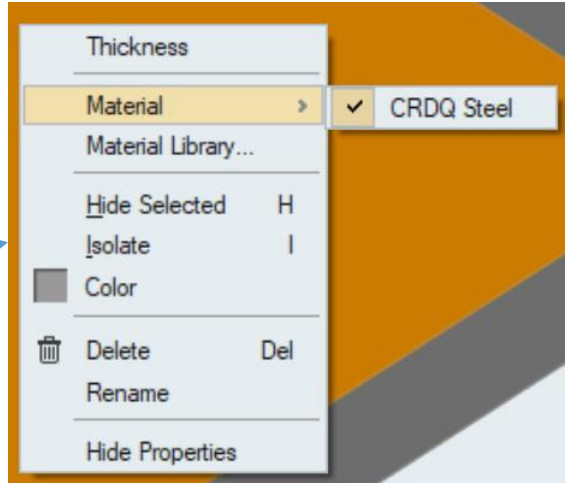
✓ **Tryout**

Material Database(1) Material DB

- **Create and Edit Materials Based on the System Library**
  - Create a material that is based on an existing predefined material in the System Library.

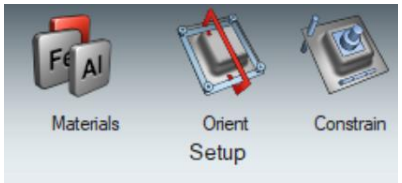


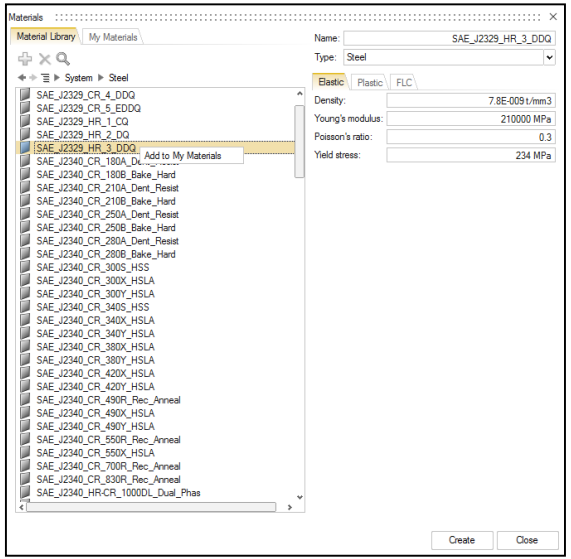


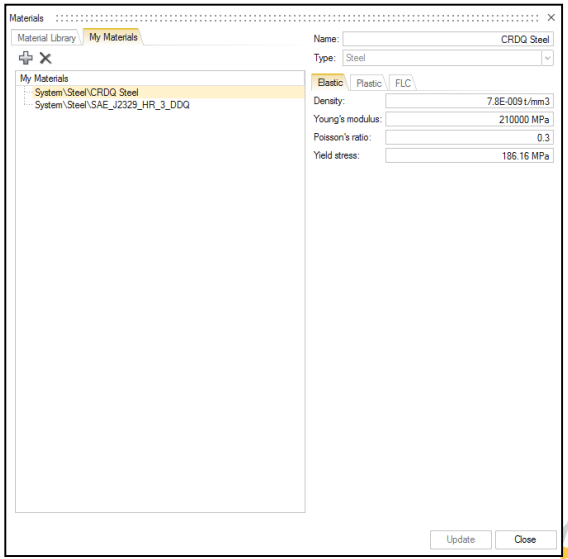


Material Database(1) Material DB

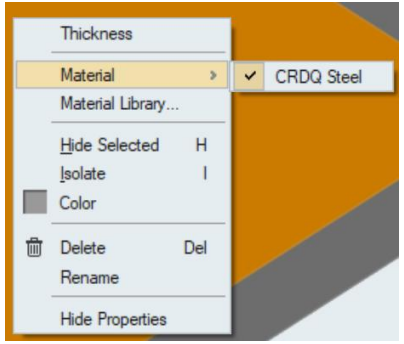
- **Create and Edit Materials Based on the System Library**
  - **Feasibility Ribbon > Setup Tool > Materials > Material Library Tab > Select the material**
  - Click Materials ribbon under Set Up to open the material library
  - Select a material from the material library and right click to add to My Materials
  - Added material can be accessed by right mouse click on the part and assigned

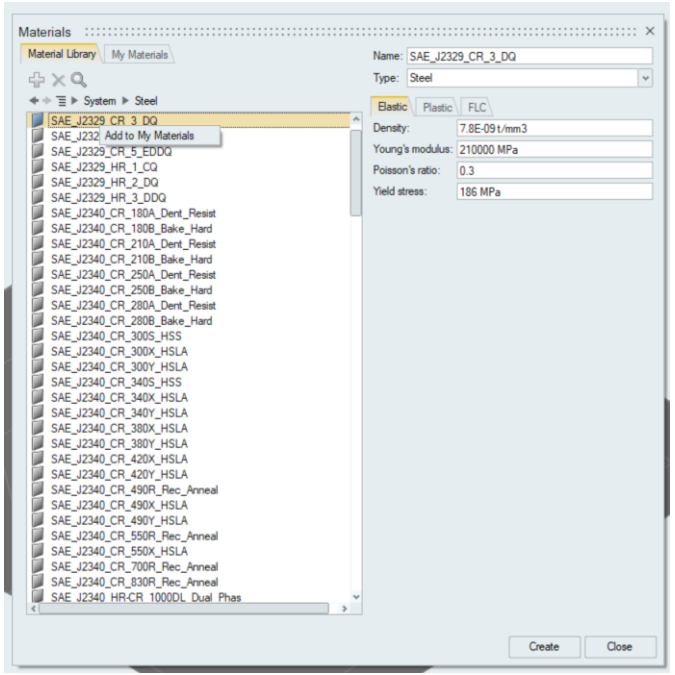


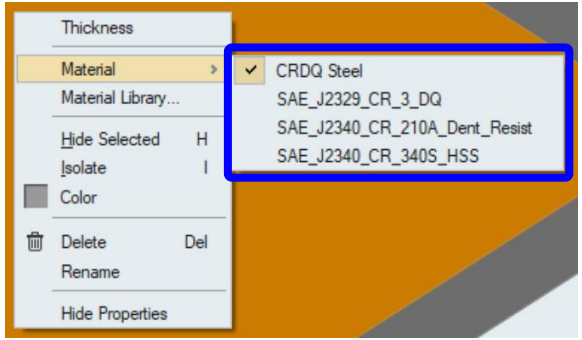




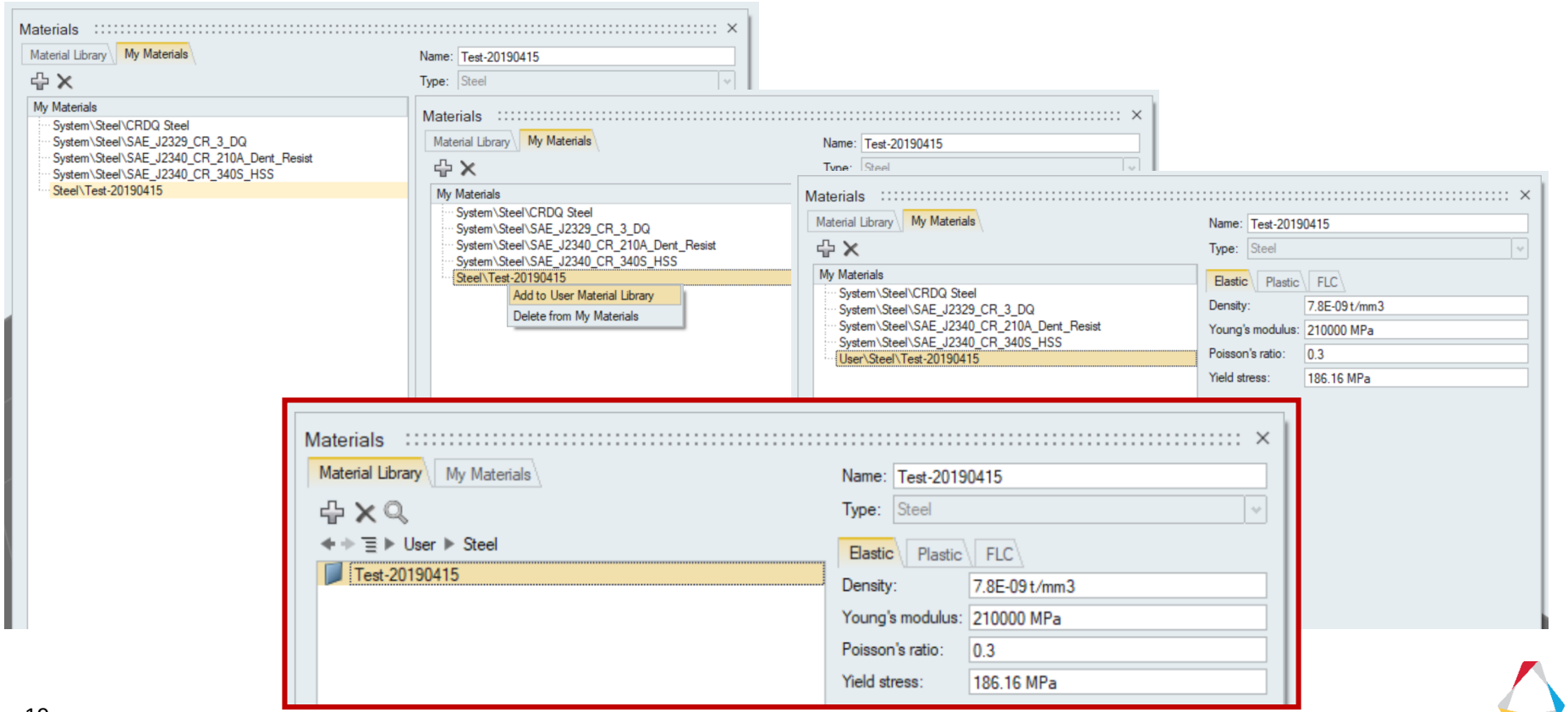
Material Database(1) Material DB



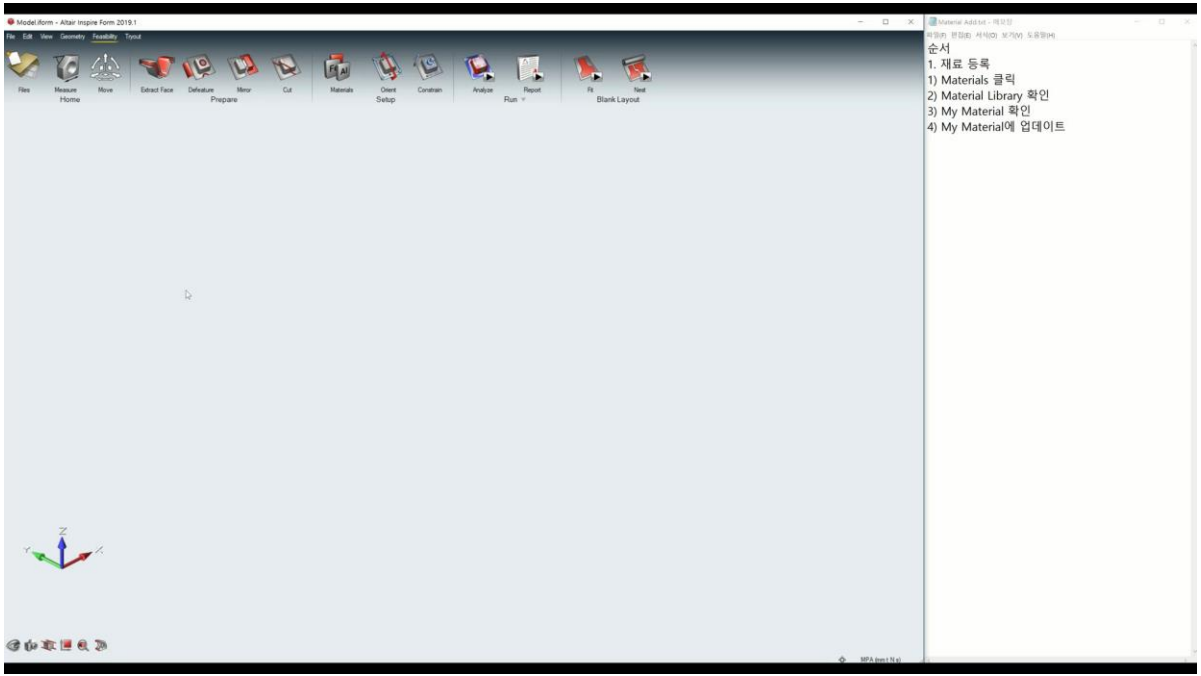




Material Database(1) Material DB



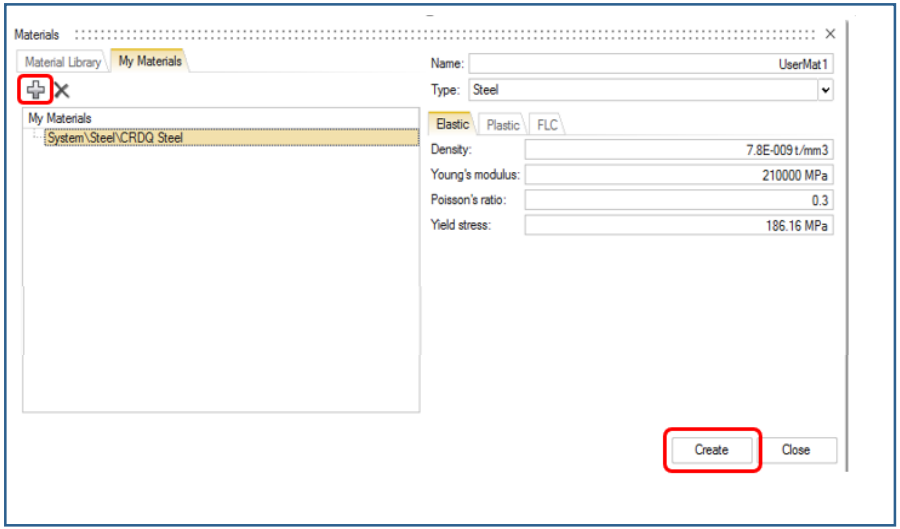
Material Database(1) Material DB Demo



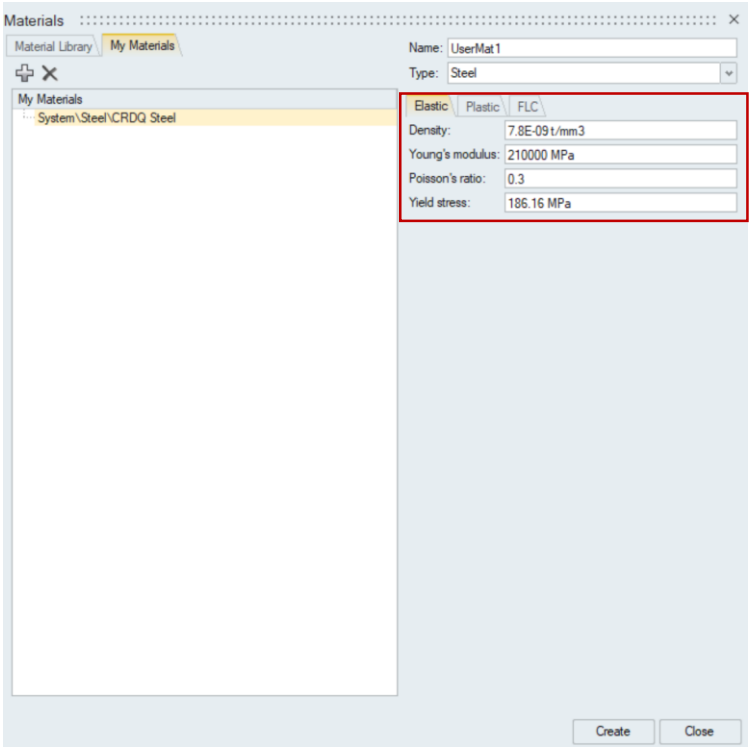
▸ 재생 시간: 2분 49초

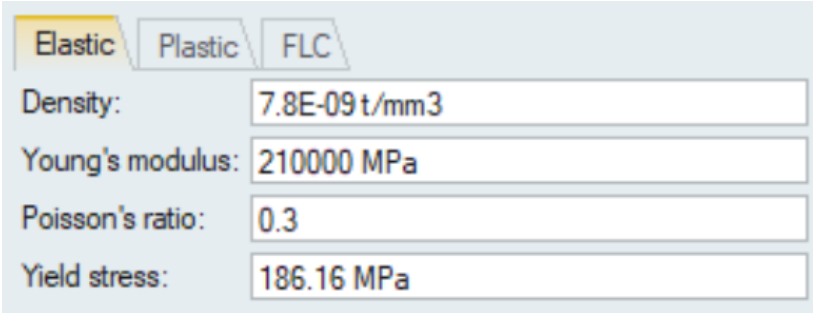
Material Database(2) Material Create

- **Create and Edit Materials from “My Materials” tab**
  - Also user can create a material under **My Materials tab** and assign to part
  - Click on the symbol  and Create button to create a new material
  - Enter a **Material Name**
  - Enter the **Elastic, Plastic** and **FLC** data and click **Create**
  - After creating the material user can also update the values at any time

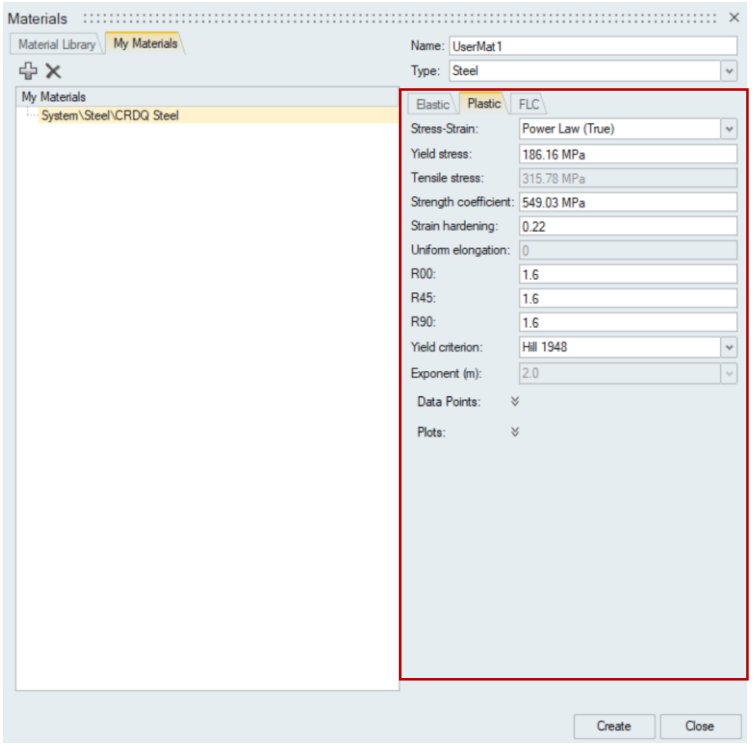


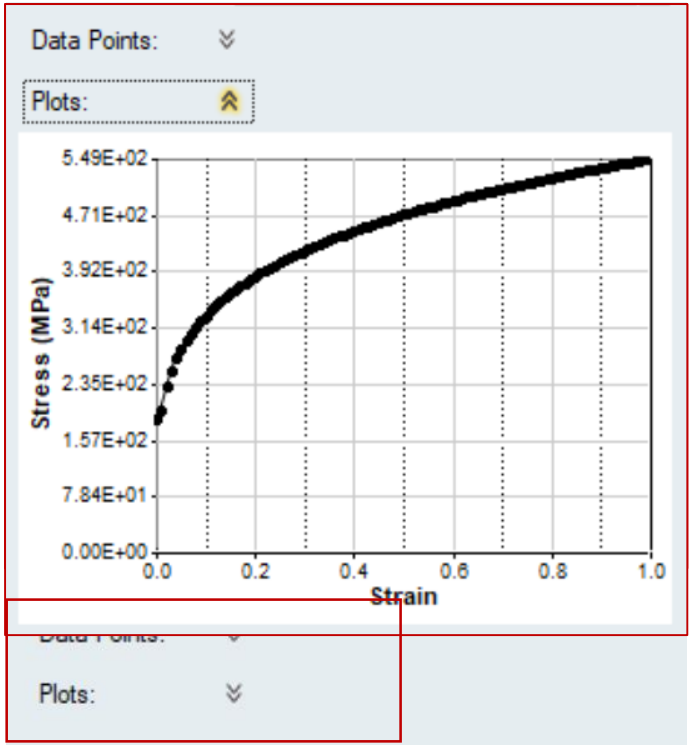
Elastic





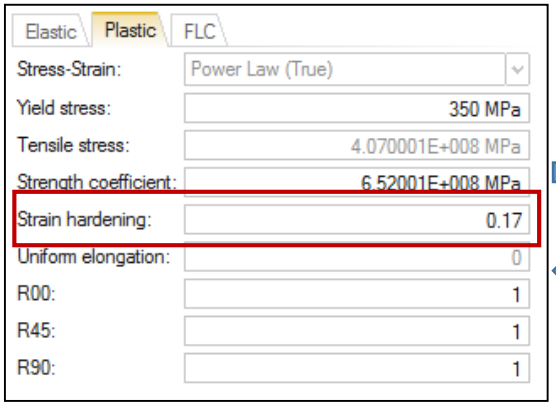
Plastic



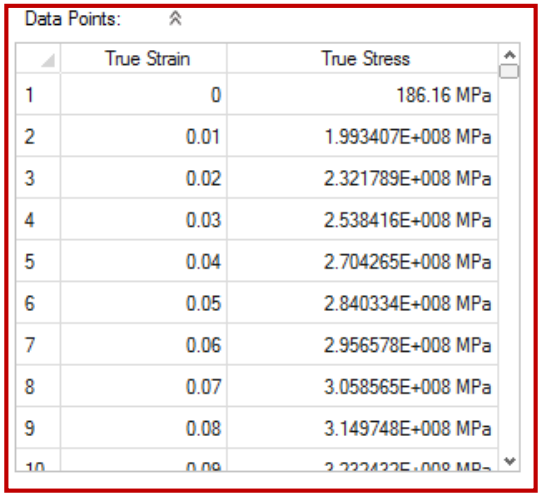


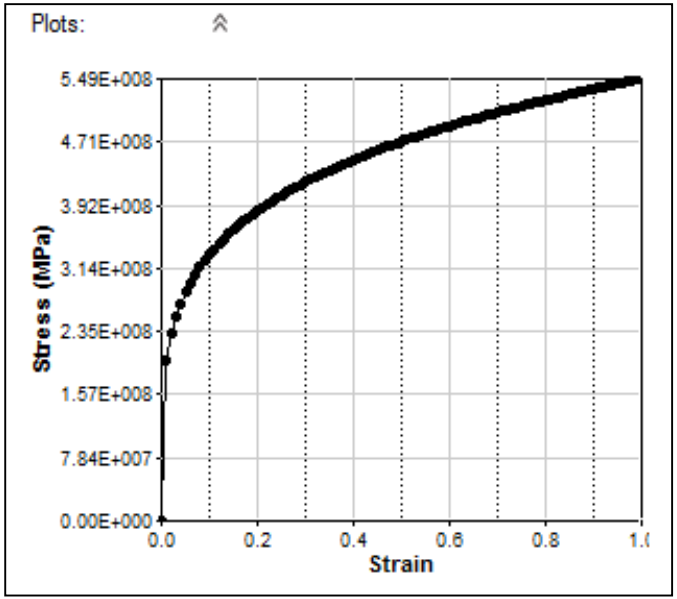
Material Database(2) Material Create

- **Create and Edit Materials from “My Materials” tab**
  - User can enter the **Plastic data** or **Data Points** (**Engineering** or **True Stress-Strain** values based)
  - Once we enter the values and create a material **Stress Strain** plot gets updated



OR





$$\sigma = (a + b\varepsilon_p{}^n)\left(1 + c\ln\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)(1 - (T^*)^m)$$

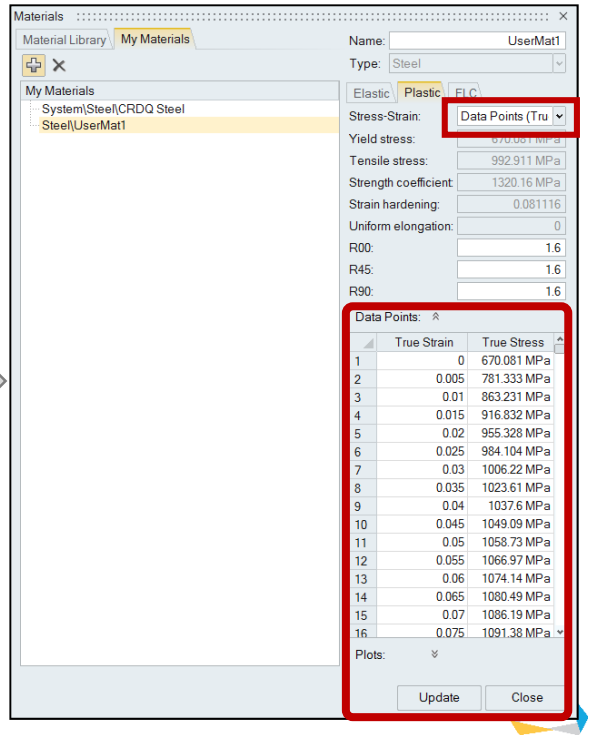
Material Database(2) Material Create

- **Create and Edit Material using Stress Strain curve data**
  - Click **My Materials** Tab inside materials window
  - Add a material by clicking on the  option
  - Load the excel file to **Data Points**

Uniaxial tension test data						
Tensile direction from RD	**Young's Modulus (GPa)**	***$S_{0.2\ (W)}$ (MPa)**	***$S_{1.0\ (W)}$ (MPa)**	**Tensile Strength (MPa)**	**Uniform Elongation (%)**	**R-Value**
0^0	186	684,2	870,8	1017,2	9,4%	0,691
15^0	187	676,9	862,6	NA	NA	0,760
30^0	186	669,9	854,5	NA	NA	0,888
45^0	191	676,6	863,4	1007,3	9,0%	1,050
60^0	197	685,2	873,8	NA	NA	1,080
75^0	204	691,4	886,3	NA	NA	1,020
90^0	208	701,5	898,0	1052,3	9,4%	0,959

User Table data

Plastic Strain	True stress
0	670.0813
0.005	781.3333633
0.01	863.2312874
0.015	916.8322885
0.02	955.3279273
0.025	984.1043743
0.03	1006.215309
0.035	1023.609047
0.04	1037.597375
0.045	1049.089413
0.05	1058.726585
0.055	1066.967202
0.06	1074.142058
0.065	1080.492057
0.07	1086.194154
0.075	1091.379477
0.08	1096.146098
0.085	1100.568094
0.09	1104.702016
0.095	1108.591538
0.1	1112.270797
0.11	1119.101303
0.12	1125.353035
0.13	1131.13382
0.14	1136.520361



Material Database(2) Material Create Demo

Step 1: 측정 항목 확인

Uniaxial tension test data						
Tensile direction from RD	Young's Modulus (GPa)	*$S_{0.2\ (W)}$ (MPa)	*$S_{1.0\ (W)}$ (MPa)	Tensile Strength (MPa)	Uniform Elongation (%)	R-Value
0^0	186	684,2	870,8	1017,2	9,4%	0.691
15^0	187	676,9	862,6	NA	NA	0.760
30^0	186	669,9	854,5	NA	NA	0.888
45^0	191	676,6	863,4	1007,3	9,0%	1.050
60^0	197	685,2	873,8	NA	NA	1.080
75^0	204	691,4	886,3	NA	NA	1.020
90^0	208	701,5	898,0	1052,3	9,4%	0.959

Plastic Strain	True stress
0	670.0813
0.005	781.3333633
0.01	863.2312874
0.015	916.8322885
0.02	955.3279273
0.025	984.1043743
0.03	1006.215309
0.035	1023.609047
0.04	1037.597375
0.045	1049.089413
0.05	1058.726585
0.055	1066.967202
0.06	1074.142058
0.065	1080.492057
0.07	1086.194154
0.075	1091.379477
0.08	1096.146098
0.085	1100.568094
0.09	1104.702016
0.095	1108.591538
0.1	1112.270797
0.11	1119.101303
0.12	1125.353035
0.13	1131.13382
0.14	1136.520361

Material Database(2) Material Create Demo

Step 2

a) Click **My Materials** Tab inside materials window

b) Add a material by clicking on the option

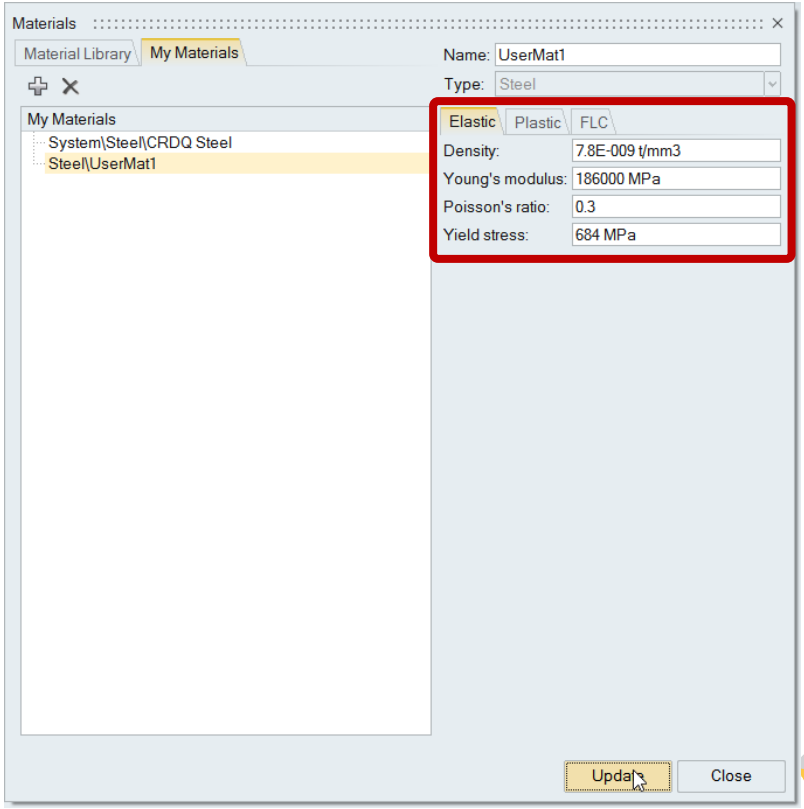
c) Enter **Name** and choose **Type.**

d) Use **Elastic** tab to fill Elastic material data. By default it is populated with steel data.

e) Define **Youngs Modulus** as 186000 Mpa.

f) Define **Yield stress** as 684 Mpa.

g) Click **Create**



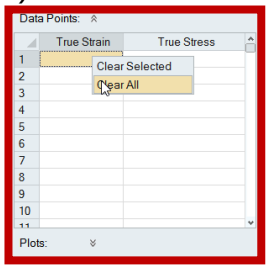
Material Database(2) Material Create Demo

Step 3

a) Enter **Plastic** Tab

b) Set **Stress – Strain:** to Data points (True)

c) Open **Data Points,** Select a field, Right click and **Pick Clear All** to empty table**.**



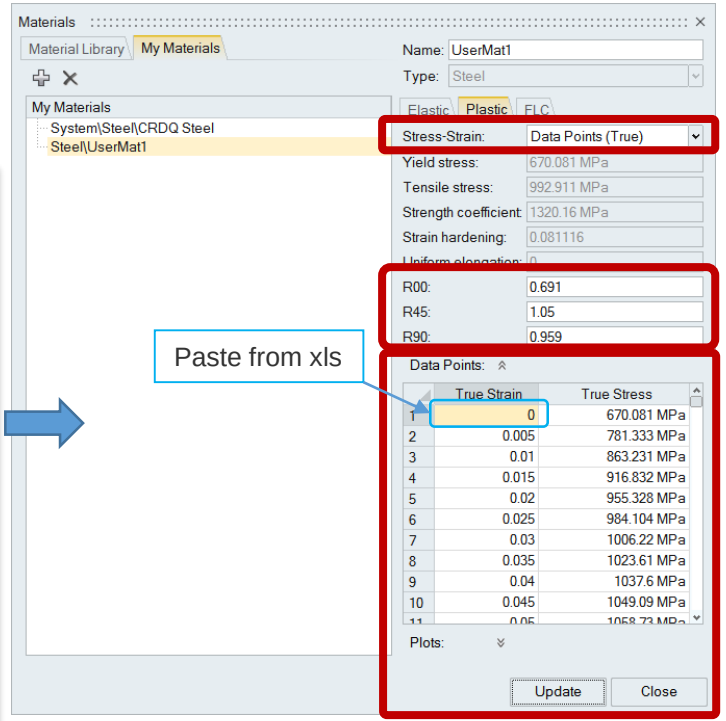
d) **Copy Data** (Ctrl+C) from excel file (data from 2 colomns), paste (Ctrl+V) in the **Data Points** selecting the first field.

Plastic Strain	True stress (Pa)
0	670081300
0,005	781333363,3

e) Fill in data for **R0, R45, R90** from the excel file

User Table data

Plastic Strain	True stress (Pa)
0	670081300
0.005	781333363.3
0.01	863231287.4
0.015	916832288.5
0.02	955327927.3
0.025	984104374.3
0.03	1006215309
0.035	1023609047
0.04	1037597375
0.045	1049089413
0.05	1058726585
0.055	1066967202
0.06	1074142058
0.065	1080492057
0.07	1086194154
0.075	1091379477
0.08	1096146098
0.085	1100568094
0.09	1104702016
0.095	1108591538
0.1	1112270797
0.11	1119101303
0.12	1125353035
0.13	1131133820
0.14	1136520361
0.15	1141569748
0.16	1146326115
0.17	1150824639
0.18	1155094031
0.19	1159158171



Material Database(2) Material Create Demo

Step 4

a) Enter **FLC** Tab.

b) Set the **FLC** to Calculate.

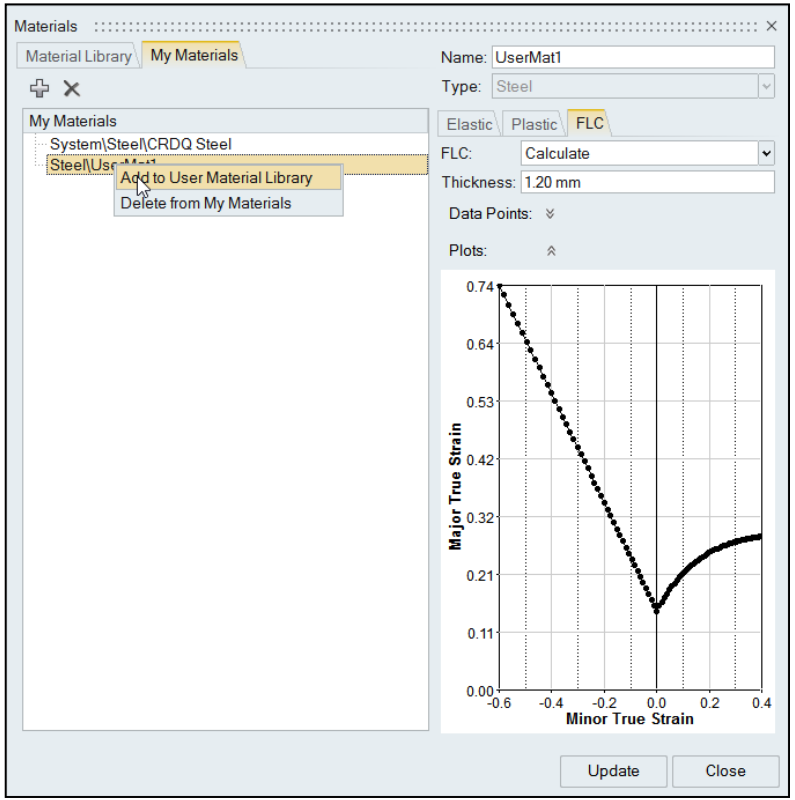
c) Enter Thickness as 1.2 mm.

d) Click on **Update** .

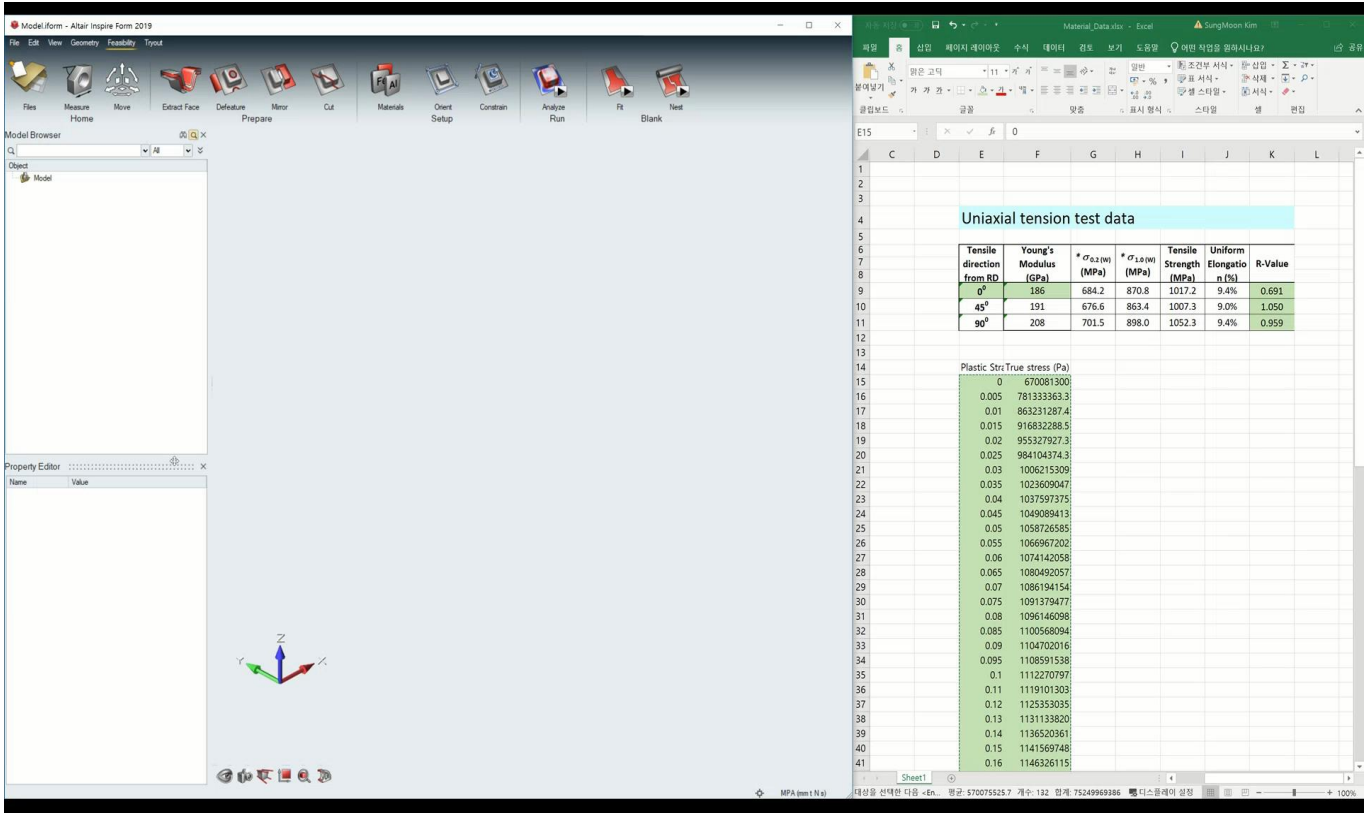
e) Select material, Right Click and Pick **Add to user Material Library** to save the material to your library (optional).

f) Click **Close** to dismiss the material dialog

$$FLC0 = (0.23 + 0.146 * t) * n / 0.21)$$



Material Database(2) Material Create Demo Video



▸ 재생시간: 2분 21초

QUESTIONS & ANSWERS